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Book Name: *Fundamentals*

1) analysis *P*

Engineering Information

There are several options in Pro/ENGINEER that provide the means for adding engineering information (material, units) and analyzing the model (measuring, interference checks, mass properties, surface analysis). This chapter describes the methods for using engineering information in the Pro/ENGINEER model, except for part differences, surface curvature analysis, and curvature analysis. These are described in Part Information in the Part Modeling User's Guide.

Topic

The Info Menu Options
Mass Property Computations
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The Info Menu Options

The **Info** option under the Main menu can be used to compute mass properties, measure distances and clearances, and obtain model information. The options available are as follows:

- **Process Info** (Process mode only)-Display information related to the process assembly (refer to the *Pro/PROCESS for Assemblies User's Guide*).

- **Mass Props**-Calculate mass properties for a model (refer to Mass Property Computations).

- **Names**-Show names of all files in memory in this session and those stored in the current directory (refer to Object Listing).

- **Measure**-Measure lengths, angles, areas, thicknesses, interferences etc. (refer to Measure).

- **Feat Info**-Show information on a feature in the selected model (refer to Feature Information).

- **Feature List**-List the features in the selected model (refer to Feature List).

- **Model Info**-Show information about a selected model (refer to Part Information and Assembly Information).

- **Comp Info** (Assembly mode only)-Show information on how a component was assembled and

display information on its parents/children, relationships and parameters (refer to <u>Layers</u> of the *Assembly Modeling User's Guide*).

- **ParentChild**-Show information on relationships between features (refer to <u>Parent/Child Information</u>).

- **Layer Info**-show information on layers in the selected model (refer to <u>Layer Information</u>).

- **Regen Info**-Regenerates the model and gives you information about each feature (refer to <u>Regeneration Information</u> of the *Part Modeling User's Guide*).

- **BOM** (Assembly mode only)-Generate a Bill of Materials (refer to <u>Bill of Materials</u> of the *Assembly Modeling User's Guide*).

- **Geom Check**-Enables you to view and re-define a feature that has a possible error (refer to <u>Geometry Checking</u> of the *Part Modeling User's Guide*).

- **Srf Analysis**-Perform surface analysis of a part with color representation (refer to <u>Surface Curvature Analysis</u> of the *Part Modeling User's Guide*).

- **Crv Analysis**-Analyze properties of curves, with color representation (refer to <u>Curvature Analysis</u> of the *Part Modeling User's Guide*).

- **Part Diff (Part mode only)**-Compares the current part with another one on disk (refer to <u>Part Differences</u> of the *Part Modeling User's Guide*).

- **Done/Return**-Return to the Main menu.

Note:

> You can now create datum features (points, axes, planes, coordinate systems), wherever you can select them under the Info menu. If the Environment option KeepInfoDtm is checked, the system marks the model as "changed" and adds those datum features to the model. If the option is not checked, the system does not mark the model as "changed" because of these datum features, and the system erases them when you quit the Info menu. See also the configuration-file option <u>keep_info_datums</u> .

Mass Property Computations

You can compute mass properties for parts, assemblies, and cross-sections by choosing Mass Props from the Info menu.

You can set the default mass units for all new parts and assemblies that you create using the configuration file option "pro_unit_mass." The possible values for the option are: "unit_ounce," "unit_pound" (default), "unit_ton," "unit_gram," "unit_kilogram," and "unit_tonne."

Note:

> Mass properties can be calculated using system parameters within a relation. However, mass properties do not automatically update when the model changes. You must recalculate the

mass properties to see the effect of model changes.

Part or Assembly Mass Properties

The computed mass properties for a part or assembly include:

- Volume-Total volume of the model.

- Surface area-Total surface area of the model.

- Density-For part mass property, the specified part density is listed here. For assembly mass property, this provides the average density of its components. The density of each individual part is used in the calculation of the mass properties.

- Mass-Total mass of the model.

- Center of gravity-Location of the center of gravity with respect to a specified coordinate system. The center of gravity and coordinate system display graphically on the model.

- Inertia tensor at the specified coordinate system origin and axes. The values are computed as follows:

$$Ixx = \int_{vol} \rho(y^2 + z^2)dV$$

etc.

$$Ixy = -\int_{vol} \rho(xy)dV$$

etc.

where ρ = specified part density.

- Inertia tensor at the center of gravity and oriented along the specified coordinate system axes.

- Principal moments of inertia-Moments of inertia at the center of gravity with respect to the principal axes

$$I1 = \int_{vol} \rho(y^2 + z^2)dV$$

etc.

The principal coordinate axes, which define a new coordinate system, are labeled "1," "2," "3" instead of "x," "y," "z;" in the equation above, "y" is the coordinate along the principal axis "2," "z" - along the principal axis "3."

- Rotation matrix and rotation angles-Rotation from the coordinate axes to the principal axes in matrix and angular form. Rotation matrix is computed as follows:

$$\begin{bmatrix} \cos\theta\cos\psi & -\cos\theta\sin\psi & \sin\theta \\ \sin\varphi\sin\theta\cos\psi + \cos\varphi\sin\psi & -\sin\varphi\sin\theta\sin\psi + \cos\varphi\cos\psi & -\sin\varphi\cos\theta \\ -\cos\varphi\sin\theta\cos\psi + \sin\varphi\sin\psi & -\cos\varphi\sin\theta\sin\psi + \sin\varphi\cos\psi & \cos\varphi\cos\theta \end{bmatrix}$$

where φ is the rotation angle about the 1 axis, θ is the rotation angle about the 2 axis, and ψ is the rotation angle about the 3 axis.
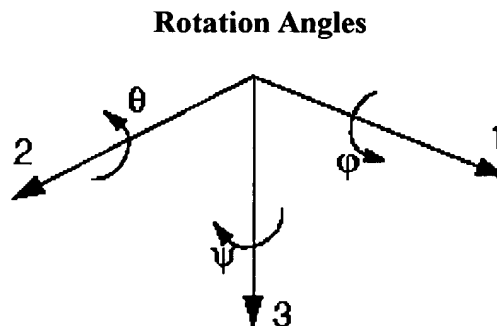
Rotation Angles

- Radii of gyration-The radii of gyration at the center of gravity with respect to the principal axes.

- For an assembly, a summary of its components' mass properties.

How to calculate mass properties for a part or assembly

1. Choose **Mass Props** from the INFO menu.

2. Choose **Part MP** or **Assembly MP** from the MASS PROPS menu.

3. If you have previously assigned mass properties to the model, the COMP/ASSIGN menu appears with the following options:

    o **Compute**-Computes the mass properties and uses these values in subsequent assemblies

    o **Assign**-Uses assigned values that have been previously set up and uses these values in subsequent assemblies.

4. If you selected **Compute or if you have not previously assigned mass properties to the model, the system prompts** you to enter the accuracy of the computation (relative to the precise analytical solution); the higher the accuracy, the longer the computations will take. Accuracy values can be entered in the range of 0.001 (1/10 of 1%) through 0.00000001 (1/10,000 of 1%).

5. The GET COORDS menu appears. Create or select a coordinate system. Mass properties can be calculated relative to the system axes (default) or relative to a user-defined coordinate system.

6. If you selected **Compute or if you have not previously assigned mass properties to the model** *and* if a part has no material assignment, the system prompts you to enter density. In an assembly, the system prompts you to enter density for each part that does not have a material assigned. However, if you had selected **Compute** but the material has already been defined for a part, Pro/ENGINEER skips this prompt and uses the defined density for the calculation. Likewise, if you had selected **Assign**, the system immediately uses the defined density (or densities) for the calculation. The proper units must be used to ensure the desired results. For example, if a part's dimensional units are inches, your density should be some mass per cubic inch. If you change the part's dimensional units, change the density to suit.

Notes:

o Part accuracy should not be less than 1/100 of the mass property accuracy.

o If dimension bounds have been set, mass property calculations are based on the hypothetical dimension values.

Typical Assembly Mass Properties File shows the contents of a typical mass properties file for an assembly.

Typical Assembly Mass Properties File

```
                    MASS PROPERTIES OF THE PART HOOK2


                VOLUME =   3.6866883e+07 INCH^3
          SURFACE AREA =   1.3798911e+06 INCH^2
               DENSITY =   2.8300000e-01 POUND / INCH^3
                  MASS =   1.0433328e+07 POUND


          CENTER OF GRAVITY with respect to REF coordinate frame:
X   Y   Z          -9.8323417e+01  1.1373033e-04 -5.5884736e+02   INCH


        INERTIA WITH RESPECT TO REF COORDINATE FRAME:   (POUND *
INCH^2)


INERTIA TENSOR:
Ixx Ixy Ixz         5.1072019e+12  2.2559395e+05 -6.2686107e+11
Iyx Iyy Iyz         2.2559395e+05  5.8895376e+12  2.3486779e+04
Izx Izy Izz        -6.2686107e+11  2.3486779e+04  7.9755144e+11


        INERTIA AT CENTER OF GRAVITY WITH RESPECT TO REF COORDINATE
FRAME:
                                            (POUND * INCH^2)
INERTIA TENSOR:
Ixx Ixy Ixz         1.8487654e+12  1.0892478e+05 -5.3572846e+10
Iyx Iyy Iyz         1.0892478e+05  2.5302370e+12 -6.3963354e+05
Izx Izy Izz        -5.3572846e+10 -6.3963354e+05  6.9668730e+11
```

```
                    PRINCIPAL  MOMENTS  OF  INERTIA:     (POUND - INCH^2)
I1    I2    I3        2.5302370e+12   1.8512513e+12   6.9420147e+11
            ROTATION  MATRIX  from  REF  coordinate  frame  to  principal  axes:
                          0.99893              0.00000              0.04635
                         -0.00000              1.00000              0.00000
                         -0.04635             -0.00000              0.99893
                 ROTATION  ANGLES  from  REF  to  principal  axes  (in  degrees):
angles  about  x    y    z      0.000             2.657                0.000
                    RADII  OF  GYRATION  with  respect  to  PRINCIPAL  AXES:
R1    R2    R3        4.9245797e+02   4.2123190e+02   2.5794751e+02      INCH
```

Note:

The values given in the summary for top-level components in the assembly mass properties output are in the units of the assembly. The densities there are with respect to volume.

Cross-Section Mass Properties

The computed mass properties for a cross-section include:

- Area-Area of the cross-section.

- Center of gravity-Location of the center of gravity with respect to a specified coordinate system. The center of gravity and coordinate system displays graphically on the model.

- Inertia tensor at the specified coordinate system origin and axes.

- Inertia tensor at the center of gravity and oriented along the specified coordinate system axes.

- Principal area moments of inertia-Moments of inertia at the center of gravity with respect to the principal axes

$$I1 = \int_{area} y^2 dS$$

$$I2 = \int_{area} x^2 dS$$

The principal coordinate axes are labeled "1" and "2" instead of "x" and "y;" in the equations above,

"x" is the coordinate along the principal axis "1," "y"-along the principal axis "2."

- Polar moment of inertia-Cross-sectional polar moment of inertia at the center of gravity.

$$Iz = \int_{area} (x^2 + y^2)dS$$

- Rotation matrix and rotation angle-Rotation around the Z-axis from the coordinate axes to the principal axes, in matrix and angular form.

- Radii of gyration-The radii of gyration at the center of gravity with respect to the principal axes.

- Section moduli and the corresponding points-Calculated for each principal axis (e.g., 1-axis, 2-axis) by dividing the area moment of inertia with respect to the chosen axis by the distance to the point(s) of the cross-section which are furthest from the chosen axis. These points have the lowest (most negative) and highest (most positive) coordinate values along the orthogonal principal axis in the cross-section plane.

For example, the section moduli about the 1-axis are the quantities derived by dividing the area moment of inertia with respect to the 1-axis by the distance to the point(s) of the cross-section which are furthest from the 1-axis. They have the lowest and highest coordinates along the 2-axis. In Typical Cross-Section Mass Properties File, the section moduli for the 1-axis are computed by searching the cross-section for the points with lowest (-2.1) and highest (.93) 2-axis coordinates, and then calculating: 2.07471e+00 INCH^3 = 4.2976190e+00 INCH^4 / 2.1e+00 INCH, and 4.62821e+00 INCH^3 = 4.2976190e+00 INCH^4 / 9.3e-01 INCH. The calculations of the section moduli for the 2-axis are analogous. The points used for the moduli calculations are defined relative to the principal axes and highlight in the graphics window.

Note:

Cross-section mass properties are not available for offset cross-sections.

How to calculate mass properties for a cross-section

1. Choose **Mass Props** from the INFO menu.

2. Choose **X-section MP** and select the cross-section name from the XSEC NAMES namelist menu.

3. Enter the relative accuracy. You can specify the accuracy of the computation (relative to the precise analytical solution); the higher the accuracy, the longer the computations take. Accuracy values can be entered in the range of 0.001 (1/10 of 1%) and 0.00000001 (1/10,000 of 1%)

4. Create or select a coordinate system. Mass properties can be calculated relative to the system axes (default) or by a user-defined coordinate system

Note:

For cross-section mass properties, the X- and Y-axes of the coordinate system must lie in a plane parallel to or coincident with the cross-section plane.

Typical Cross-Section Mass Properties File shows the contents of a typical cross-section mass-properties file.

Typical Cross-Section Mass Properties File

```
 MASS PROPERTIES OF THE CROSS SECTION XSECT_PLANAR
 AREA =  2.8267949e+04 INCH^2
 CENTER OF GRAVITY with respect to _XSECT_PLANAR coordinate frame:
 X    Y                    -6.1272602e+00 -5.1838178e+01   INCH
 INERTIA with respect to _XSECT_PLANAR coordinate frame:   (INCH^4)
 INERTIA TENSOR:
 Ixx Ixy                    9.9263878e+07 -3.4641016e+06
 Iyx Iyy                   -3.4641016e+06  2.0623612e+08
 POLAR MOMENT OF INERTIA:                 3.0549999e+08 INCH^4
 INERTIA at CENTER OF GRAVITY with respect to _XSECT_PLANAR coordinate
 frame :
                                                                 (INCH^4
 )
 Ixx Ixy                    2.3302338e+07  5.5145342e+06
 Iyx Iyy                    5.5145342e+06  2.0517484e+08
 AREA MOMENTS OF INERTIA with respect to PRINCIPAL AXES:   (INCH^4)
 AREA MOMENTS OF INERTIA with respect to PRINCIPAL AXES:   (INCH^4)
 POLAR MOMENT OF INERTIA:                 2.2847718e+08 INCH^4
 INERTIA TENSOR:
 ROTATION MATRIX from _XSECT_PLANAR orientation to PRINCIPAL AXES:
 0.99954          0.03028
 -0.03028         0.99954
 ROTATION ANGLE from _XSECT_PLANAR orientation to PRINCIPAL AXES
 (degrees) :
 RADII OF GYRATION with respect to PRINCIPAL AXES:
 R1  R2                     2.8608184e+01  8.5229832e+01   INCH
 SECTION MODULI and corresponding points:
 MODULUS                            1         2    COORD
 about AXIS 1:   4.40705e+05 INCH^3         -1.4235e+02 -5.2496e+01 INCH
 4.09171e+05 INCH^3           1.5449e+02  5.6542e+01 INCH
 about AXIS 2:   1.41248e+06 INCH^3         -1.4538e+02  4.7458e+01 INCH
 1.30364e+06 INCH^3           1.5751e+02 -4.3412e+01 INCH
```

Effects of Suppressed Features and Blanked Parts

Suppressed features: If features are suppressed, the mass properties will be calculated as if the features did not exist. If a part has been simplified for assembly purposes and contains many suppressed features, mass properties may be assigned to the part so that those values are used in calculating mass properties of an assembly. *See the next section for information on assigning mass properties.*

Blanked parts: If parts on layers have been blanked from the current view, they are still used to calculate the mass properties of an assembly. Blanking items only affects the display of the object, not its composition.

Calculating the Multiple Volumes

How to calculate the volume to one side of a datum plane or pattern of datums

1. In Part mode, choose **Info, Mass Props**, then **OneSidedVols**.

2. Enter a relative accuracy for the part.

3. Select a datum plane. If the datum plane is patterned, the whole pattern is used.

4. Select which side to calculate the volumes for, using the flip arrow.

5. The volumes appear in an info window, and are written to a file "*partname*.osv."

In the output information, offset values given for the datums are from the selected datum, and the datum plane that was picked has
"Selected" next to it.

Angular datums can also be used, but no offset dimension value appears in the output.

Object Listing

The Info menu option Names displays a listing of object files in an Information Window. The first portion of the listing shows parts, assemblies, drawings, layouts and sections in memory. The second portion gives a complete listing of all Pro/ENGINEER objects in the working directory.

Measure

The Measure command in the menu is used to analyze model and draft geometry. It is available in all modes of operation.

Edges and Curves

Measure part edges and datum curves with the Curve/Edge option from the Measure menu. The information available through the Info Curve menu is as follows:

- **Length**-Displays, in the message window, the length of the selected edge or curve.

- **Type**-Displays, in the message window, the type of the edge or curve.

- **Normal**-Displays in red the normal vector (2nd derivative) to the edge or curve *at the selected point*. Displays the size of the dx, dy and dz elements of the normal vector in the message window. You have to select or create a reference coordinate system.

- **Tangent**-Displays in red the tangent vector (first derivative) to the edge or curve *at the selected point*. Displays the size of the dx, dy and dz elements of the tangent vector in the message window. You have to select or create a reference coordinate system.

- **Curvature**-Displays, in the message window, the curvature (1/radius) of the edge or curve *at the selected point*.

- **Radius**-Displays, in the message window, the radius (1/curvature) of the edge or curve *at the*

selected point.

- **All**-Displays, in an Information Window, the combined information provided by Length, Type, Normal, Tangent, and Curvature. The selection procedure and screen display for each edge, curve, and point are the same as with the individual options.

- **Max Dihedral**-Displays, in the message window, the maximum angle between the normals of the two surfaces that border an edge.

- **Short Edge**-Highlights in green, the edges that are shorter than a specified length. Displays the total number in the message window.

How to measure an edge or datum curve

1. Choose **Info, Measure, Curve/Edge**. The INFO CURVE menu appears.

2. Choose the appropriate option from the INFO CURVE menu.

3. If you chose **Normal, Tangent**, or **All**, select or create a coordinate system using the GET COORD S menu options:

    o **Create**-Open the Csys OPTIONS menu, so that you can create a new coordinate system "on the fly" (refer to Coordinate Systems of the *Part Modeling User's Guide*).

    o **Select**-Select an existing coordinate system.

    Go to Step 5.

4. If you chose **Curvature** or **Radius**, select the point at which to make the measurement, through the POINT OPT menu:

    o **Select**-The measurements are made for the pick point.

    o **End point**-The measurements are made for the nearest endpoint of the edge or curve segment.

5. Select an edge or curve to be measured. If you are measuring the length of several curves/edges lying on the same surface, use the **Sel Chain** command. Select the end entities of the chain and use the CHOOSE menu options to determine the chain to measure. The selected entity is highlighted in blue.

6. The information displays in the message window or in an information window.

    Note:

    Composite curves have selection priority over the separate segments: if you want to measure a curve segment instead of the whole curve, use Query Sel *or* Sel By Menu.

Max Dihedral

The Max Dihedral command in the Info Curve menu traverses a selected edge, and calculates the maximum angle between the surfaces that border it. The normals at the edge intersection are used.

**Calculating the Maximum Dihedral Angle**
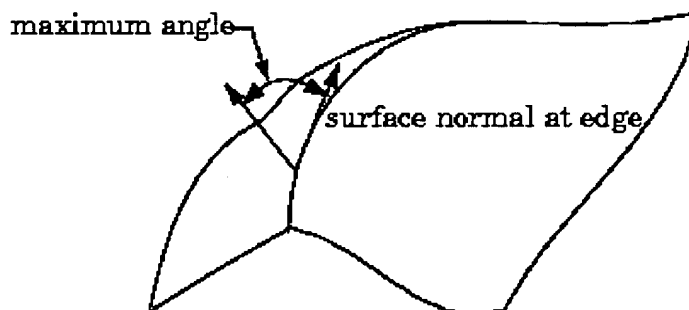
Angles

Use the command Angle to measure the angle between two entities. The entities can be axes, planar curves, or planar non-linear edges. The angle that the system measures depends on where you select the edges, as shown below.

You can also measure the i, j, k angles between a coordinate system and a curve, edge, axis or plane.

Different Angle Measurements
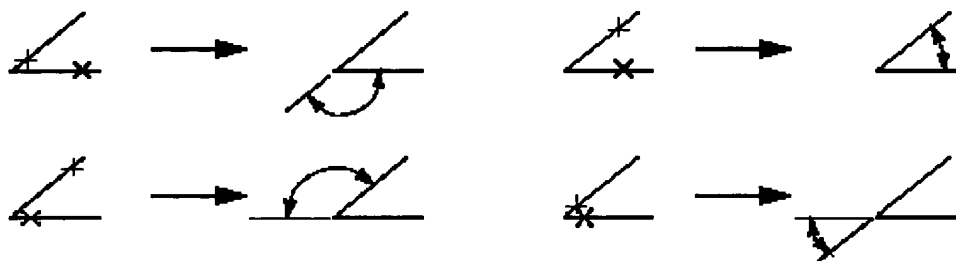
When you ask the system to measure the angle to/from a planar, non-linear edge or planar curve, the system actually calculates the angle to/from the *plane* in which the selected edge or curve lies. When the system is computing the value of the angle between two planes or between a plane and a line, it selects the *smaller* of the two possible angle values (*see the following figure*).

Computing the Angle Between a Plane and a Line

The smaller angle between DTM1 and DTM2 will be the angle between DTM1 and the curve
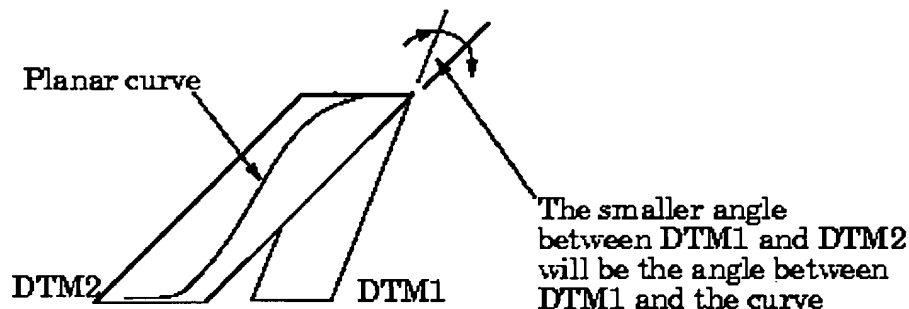
Pro/ENGINEER measures angles with respect to a basis entity. The "basis entity" is the first entity that

you select. After you have selected the basis entity, you can measure as many angles to it as you like by selecting various other entities in turn. The system calculates each angle with respect to the first entity until you restart the measuring process by selecting a new basis entity

How to measure an angle

1. Choose **Info, Measure, Angle**. The ANGLE REF menu appears with the option **First Ent** pre-selected.

2. Select the type of the first entity.

    - **Curve/Edge**-Select a planar edge or curve.

    - **Axis**-Create or select an axis. Go to Step 4.

    - **Plane**-Create or select a plane. Go to Step 5.

    - **Csys**-Create or select a coordinate system. Go to Step 6.

3. If you chose Curve/Edge, select the curve or edge, then go to Step 7.

4. If you chose Axis, the GET DATUMS menu appears with the options:

    - **Select**-Select a feature axis or a datum axis

    - **Create** - Open the DATUM AXIS menu, so that you can create a new datum axis "on the fly" (refer to <u>Datum Axes</u> of the Part Modeling User's Guide).

      Select or create an axis, then go to Step 7

5. If you chose Plane, the GET DATUMS menu appears with the options:

    - **Select**-Select a planar surface or a datum plane.

    - **Create**-Open the DATUM PLANE menu, so that you can create a new datum plane "on the fly" (*refer to <u>Datum Planes</u> of the* Part Modeling User's Guide).

      Select or create a plane, then go to Step 7.

6. If you chose Csys, the GET COORD S menu appears with the options:

    - **Create**-Open the Csys OPTIONS menu, so that you can create a new coordinate system "on the fly" (*refer to <u>Coordinate Systems</u> of the* Part Modeling User's Guide).

    - **Select**-Select an existing coordinate system.

    - **Default**-Select the default coordinate system, if it already exists. If it does *not* already exist, the system creates a "virtual" default coordinate system to make these measurements.

7. Select the **Second Ent** option from the ANGLE REF menu, then select its type.

8. The system displays the measured angle in the message window.

   Note that if one of the entities is a coordinate system, the measured angle has three elements, i, j, and k, defined as follows:

   i = the angle between the coordinate system's y-z plane (i.e. normal to the x-axis) and the other entity;

   j = the angle between the coordinate system's x-z plane (i.e. normal to the y-axis) and the other entity;

   k = the angle between the coordinate system's x-y plane (i.e. normal to the z-axis) and the other entity;

9. To measure the angle between the basis entity and another entity, return to Step 6.

10. To select a new basis entity, choose **First Ent,** then return to Step 2.

11. Choose **Done-Return** to return to the MEASURE menu.

Distance

Pro/ENGINEER measures distances with respect to a basis entity. The "basis entity" is the one from which you measure, that is, the first entity you select when you start measuring Distance. After you have selected the basis entity, you can measure as many distances from it as you like by selecting various other entities in turn. The system calculates all the distances with respect to the first entity until you restart the measuring process by selecting a new basis entity.

   Note:

   If necessary, Pro/ENGINEER extends two linear entities, in order to measure the distance between them.

The figure below shows how the Measure Distance function is used to measure the minimum distance between two curves.

Measuring Minimum Distance Between Two Curves
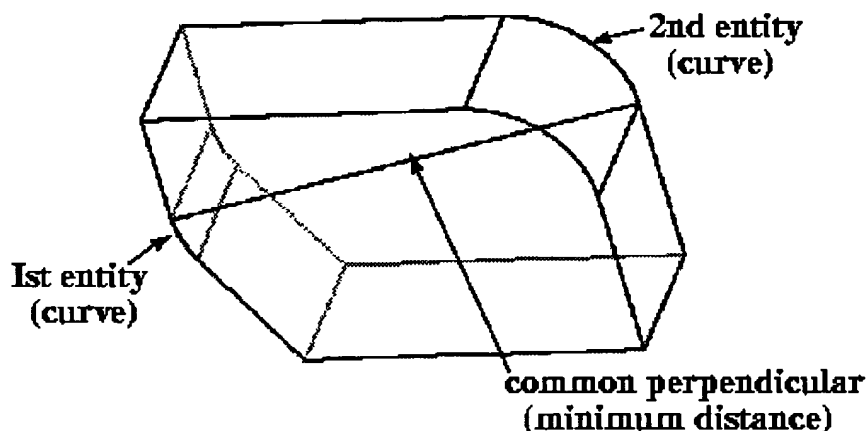
The following figure shows how the Measure Distance function extends two linear entities to find the minimum distance between them.

Measuring Minimum Distance Between Two Non-Parallel, Non-Coplanar Straight Lines
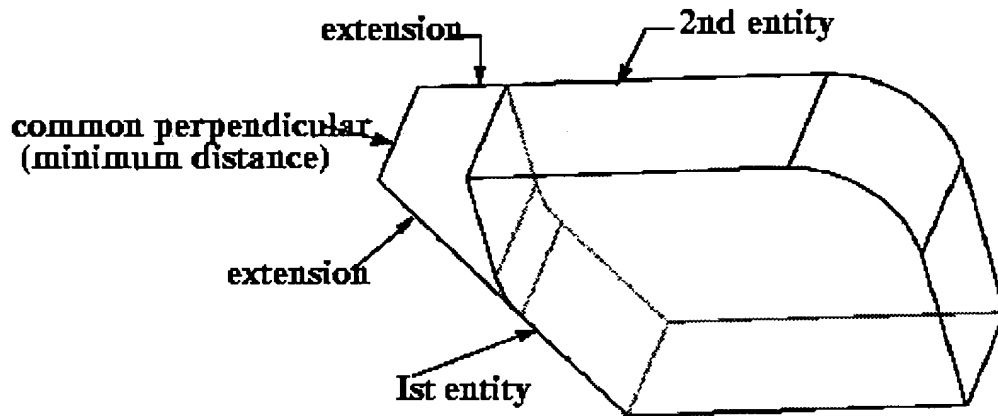
When you are measuring distances, you have to specify the entity type before selecting an entity from or to which to measure. The entity types are:

- **Point**-Any point on the part surface, or a datum point.

- **Vertex**-Any vertex of the part.

- **Plane**-Planar part surface or datum plane.

- **Lin Ent**-A linear entity, such as an axis, edge, 3D curve, pipe centerline or cable centerline.

- **Csys**-A coordinate system.

Measuring From a Coordinate System

When you choose Coord sys as the basis entity in absolute measurements, you can choose from three coordinate types: Cartesian, Cylindrical, or Spherical.

Incremental Measurements

When you are making *incremental* measurements, you must first define a reference coordinate system. then choose one of three coordinate types: Cartesian, Cylindrical, or Spherical. You must then select both "From" and "To" entities for each measurement. All the incremental measurements are made by comparing the coordinates of the two entities with respect to the reference coordinate system.

How to measure the distance between any two entities

1. Choose **Info, Measure, Distance**. The DISTANCE menu will display.

2. Choose to use absolute or incremental coordinates:

    ◦ **Absolute**-Display the measured distance between the two entities.

- **Increment**-Display *both* the measured distance between the two entities *and* the differences between their coordinates (e.g., dx, dy, dz) with respect to the reference coordinate system. Both the **From** and **To** options are inaccessible until you select or create the reference coordinate system.

  If you chose **Absolute**, go to Step 5.

3. If you chose **Increment**, the GET COORD S menu appears with the options:

   - **Create**-Open the Csys OPTIONS menu, so that you can create a new coordinate system "on the fly" (r*efer to Coordinate Systems of the* Part Modeling User's Guide).

   - **Select**-Select an existing coordinate system.

     Create or select a coordinate system.

4. The COORD TYPE menu appears. Select one of the three coordinate types-**Cartesian, Cylindrical** or **Spherical**.

5. Choose the appropriate **From** option to establish the basis entity.

   - **From Point**-Measure from an arbitrary point.

   - **From Vertex**-Measure from a vertex.

   - **From Plane**-Measure from a plane (surface or datum).

   - **From Lin Ent**-Measure from an axis, edge, 3D curve, cable centerline or pipe centerline.

   - **From Csys**-Measure from a coordinate system.

6. If you chose From Point, the GET DATUMS menu appears with the options:

   - **Select**-Pick or select a point on the model.

   - **Create**-Open the DATUM POINT menu, so that you can create a datum point "on the fly" (refer to Datum Points of the Part Modeling User's Guide).

     Select or create a point, then go to Step 10.

7. If you chose Vertex, pick a vertex on the model. Go to Step 10.

8. If you chose Plane, the GET DATUMS menu appears with the options:

   - **Select**-Select a planar surface or a datum plane.

   - **Create**-Open the DATUM PLANE menu to create a new datum plane "on the fly" (refer to Datum Planes of the Part Modeling User's Guide).

Select or create a plane, then go to Step 10.

9. If you chose Csys, the GET COORD S menu appears with the options:

   ○ **Create**-Open the Csys OPTIONS menu, so that you can create a new coordinate system "on the fly" (refer to <u>Coordinate Systems</u> of the Part Modeling User's Guide).

   ○ **Select**-Select an existing coordinate system.

     If you chose Absolute measurement in Step 2, the COORD TYPE menu appears. Select one of the three coordinate types-**Cartesian**, **Cylindrical** or **Spherical**.

10. Choose the appropriate **To** option in the same way as you chose the *From* option:

    ○ **To Point**-Measure to an arbitrary point.

    ○ **To Vertex**-Measure to a vertex.

    ○ **To Plane**-Measure to a plane (surface or datum).

    ○ **To Lin Ent**-Measure to an axis, edge, 3D curve, cable centerline or pipe centerline.

    ○ **To Csys**-Measure to a coordinate system.

11. Make as many measurements as you want from the basis entity. You can change the type of entity to which you are measuring at any time.

12. To restart the measuring process for a new basis entity, repeat the process from Step 2, choosing a new basis entity option (i.e., **From...**).

13. Choose **Done/Return** from the DISTANCE menu to return to the MEASURE menu.

    Note:

      In Assembly, all measurements are based on the unexploded assembly distances (exploding an assembly only affects the *view* of assembly components).

Tolerance Stackup

If dimension bounds have been set, then measurement calculations are based on new dimension values.

How to calculate tolerance stackup

1. Set the dimension bounds (*refer to <u>Dimension Bounds</u>*).

2. Measure **Distance** between the desired entities.

Clearance and Interference Calculations

The Clear/Intf option calculates and displays either the clearance distance or interference between any

combination of sub-assemblies, parts, surfaces, cables, or entities. Since an exploded assembly is only a modification of the displayed view, calculations are not affected by explosion distances.

Note that calculation accuracy is determined by part accuracy; the accuracy of a clearance measurement or interference volume is controlled by the configuration file option "measure_sig_figures."

If the objects selected do not interfere, the minimum clearance displays graphically as a red line. A small red circle with cross-hair displays at each end of the line to identify the location at which the clearance is being measured. The clearance value displays in the Message Window. If there is interference between the two surfaces, the system highlights the volume of interference and provides the value or highlights the curve or point of intersection, as appropriate for the items selected.

The Pairs option enables you to select the two objects or entities for calculations while two global options automatically perform the check on all pairs in the assembly.

The Volume Intf option is used in ECAD to ensure that keepin/keepout areas have not been violated.

How to determine a generic clearance or interference

1. Choose **Info, Measure, Clear/Intf**. The CLEAR/INTF menu appears.

2. Choose the desired option from the CLEAR/INTF menu. The CLEAR/INTF options are:

   o **Pairs**-Get clearance or volume of interference between pairs of any combination of sub-assemblies, parts, surfaces, cables, and/or entities. This causes the PAIRS, SELECT_TYPE, PAIR CALC, and GET SELECT menus to display.

   o **Global Clr**-Find all pairs of parts or sub-assemblies that have clearances less than a specified clearance distance.

   o **Global Intf**-Find all interfering pairs of parts or sub-assemblies.

   Note:

   If dimension bounds have been set, clearance/interference calculations are based on new dimension values.

Clearance/Interference Between Pairs

How to determine the clearance distance or the amount of interference between a pair of any combination of sub-assemblies or cables, parts, surfaces, and/or entities

1. Choose **Pairs** from the CLEAR/INTF menu. The PAIRS menu appears with the **First** option pre-selected.

2. The SELECT_TYPE menu appears Choose from one of the options for the first selection. They are as follows:

   o **Whole Subasm**-Use an entire sub-assembly in the clearance/interference check, but do not check between components within the sub-assembly.

- ○ **Whole Part**-Use an entire part in the clearance/interference check.

- ○ **Surface**-Use a surface in the clearance/interference check. Go to Step 2 of <u>Clearance/Interference Between Surfaces</u> below, to continue this process.

- ○ **Cable**-Use an entire cable feature in a harness part in the clearance/interference check.

- ○ **Single Ent**-Use a single entity in the clearance/interference check.

3. If you chose **Whole Subasm** or **Whole Part** in Step 2, choose a PAIR CALC menu option:

- ○ **ExcludQuilts**-Compute interference between solids only. Note that if a part or sub-assembly consisting only of quilts is selected, the calculations cannot be performed.

- ○ **IncludQuilts**-Take into account visible quilts when computing interference.

4. Pick the first object.

5. If desired, choose a different SELECT_TYPE option for the second selection.

6. Pick the second object in the pair. The **Second** button of the PAIRS menu stays selected until you either choose **Done/Return** in the CLEAR/INTF menu or re-choose **First** in the PAIRS menu. This enables you to continue to use any SELECT_TYPE option for multiple second picks, each of which is measured from the first selection unless you re-choose **First**. Rechoosing **First** enables you to choose a new first reference for subsequent clearance or interference calculations by returning to step 2.

7. When the process is complete, choose **Done/Return** in the CLEAR/INTF menu.

Clearance/Interference Between Surfaces

How to find the clearance using surfaces

1. Choose **Pairs** from the CLEAR/INTF menu.

2. Choose **Surface** from the SELECT_TYPE menu. The SURF CLR menu appears with the options:

- ○ **Whole Surf**-Measure the minimum distance anywhere between the selected surface and the second object/entity selected.

- ○ **Near Pick**-Measure the minimum distance between the selected surface and a second surface at a pick point on each. Only surfaces may be selected as the second item if **Near Pick** has been selected for the first surface.

3. Pick a surface on the model.

4. Select a second surface of the model.

5. The minimum clearance or line of intersection displays, as appropriate. The clearance value

displays in the Message Window.

Global Clearance within an Assembly

How to identify in an entire assembly all component parts or sub-assemblies for which clearances are less than or equal to a clearance distance that you specify

1. Choose **Global Clr** from the CLEAR/INTF menu. The GLOBAL SETUP menu appears, with its options grouped in mutually-exclusive pairs. Choose the desired options and **Done/Return**.

   Pair 1:

   - **Subasms Only** -Do a global check for clearances between *all sub-assemblies*, but not within individual sub-assemblies.

   - **Parts Only**- Perform a global check for clearances between *all parts* in the assembly.

   Pair 2:

   - **ExcludQuilts**- Compute clearances between solids only.

   - **IncludQuilts**- Not available for global clearance processing.

2. Enter the clearance distance, as prompted in the Message Window. The system determines if any components of the assembly are within the specified clearance distance. All interferences is included.

3. Use **Next** or **Previous** in the GLOBAL CLR menu to step through the display of identified pairs. To exit the process, choose **Done/Return** in the CLEAR/INTF menu.

Global Interference within an Assembly

How to check an entire assembly for only those component parts or sub-assemblies which interfere

1. Choose **Global Intf** from the CLEAR/INTF menu. The GLOBAL SETUP menu appears, with its options grouped in three mutually- exclusive pairs. Choose the desired options and **Done/Return**.

   Pair 1:

   - **Subasms Only**-Do a global check for interferences between all sub-assemblies, but *not* within individual sub-assemblies.

   - **Parts Only**-Do a global check for interferences between *all* parts in the assembly.

   Pair 2:

   - **ExcludQuilts**-Compute the amount of interference between solids only.

   - **IncludQuilts**-Take visible quilts into account when computing the amount of interference. Merged surfaces are considered during the interference calculation, and areas where they

interfere highlight.

Pair 3:

- o **Exact Result**-Perform high-precision calculation. Provide a list of all interfering pairs of parts or subassemblies and the interference volume in each pair.

- o **Quick Check** -Perform a first-pass check to save time. Provide a list of all interfering pairs of parts or subassemblies.

2. The system finds all interfering parts in the assembly, calculates the amount (volume) of interference, and then displays interfering parts, if any. The system highlights Interfering parts one pair at a time, highlighting one part in blue and the other in yellow. The interference volume highlights in red.

Excluding Quilts

1. If you chose **ExcludQuilts** in Step 1, use the SOLID INTERF menu options to step through the assembly:

- o **Nex**-Display the next pair of interfering parts. If there are none, the option is dimmed.

- o **Previous**-Display the previous pair of interfering parts. If there are none, the option is dimmed.

- o **Verify**-Calculate the volume of the interference region of the highlighted pair. The region highlights in red on the model, and its volume displays in the message window.

Including Quilts

1. If you chose **IncludQuilts** in Step 1, use the QUILT INTERF menu options to step through the assembly:

- o **Nex**-Display the next pair of interfering parts. If there are none, the option is dimmed.

- o **Previous**-Display the previous pair of interfering parts. If there are none, the option is dimmed.

2. When the process is complete, choose **Done/Return** in the CLEAR/INTF menu.

When the system checks global interference for exploded assemblies, it highlights interfering parts in their exploded positions, one part in red and another in yellow. When **Verify** is selected, the interference area displays for each part, in the color of that part (red or yellow).

Surface Area

The MEASURE command **Area** measures the area of any surface on the part or the area of a quilt or the area of a datum surface.

How to measure the area of a surface

1. Choose **Info, Measure, Area**. The AREA menu appears with its options grouped in two sets as follows:

   Set 1.

   - o **Actual**-Calculate the actual area of a surface or quilt.

   - o **Projected**-Specify a projection direction and calculate the area of a surface or quilt projected in that direction.

   Set 2.

   - o **Surface**- Calculate the surface area of a face.

   - o **Quilt**- Calculate the surface area of a quilt.

2. If you chose Actual from the first set, go to Actual Area.

3. If you chose Projected from the first set, go to Projected Area.

Actual Area

How to measure the "Actual" area of a surface

1. If you chose Actual from the first set in the AREA menu, then choose either **Surface** or **Quilt** from the second set.

2. Select a part surface or a quilt, as appropriate, for area measurement. The system highlights the selected entity in red, and displays the area in the message window.

   Note:

   When you are selecting a cylindrical or conical shaped surface, remember that you can select only half of the surface at a time.

Projected Area

How to measure the "Projected" area of a surface

1. If you selected Projected from the first set in the AREA menu, the system prompts you to define the direction of projection by selecting a curve, edge or axis or by selecting a plane normal to the direction of projection. The GEN SEL DIR menu appears with these options:

   - o **Plane**-Select a plane normal to the direction of projection.

   - o **Crv/Edg/Axis**-Use a curve, edge or axis to define the direction of projection.

   - o **Csys**-Use an axis of a coordinate system as the direction of projection.

Select one of the options.

2. If you chose Plane in Step 1, the GET DATUMS menu appears with the options:

   ○ **Select**-Select a planar surface, a quilt or a datum plane.

   ○ **Create**-Open the DATUM PLANE menu, to create a new datum plane "on the fly" (refer to Datum Planes of the Part Modeling User's Guide).

   Select or create a plane, then go to Step 5.

3. If you chose Crv/Edg/Axis in Step 1, the GET DATUMS menu appears with the options:

   ○ **Select**-Select a curve, edge or axis. If you select a curve or an edge that is not linear, the system prompts you to specify a tangent by selecting a previously-defined datum point on the entity

   ○ **Create**-Open the DATUM AXIS menu, to create a new datum axis "on the fly" (refer to Datum Axes of the Part Modeling User's Guide).

   Select or create an entity of this type, then go to Step 5.

4. If you chose Csys in Step 1, the GET COORDS menu appears with the options:

   ○ **Select**-Select an existing coordinate system.

   ○ **Create**-Open the Csys OPTIONS menu, so that you can create a new coordinate system "on the fly" (refer to Coordinate Systems of the Part Modeling User's Guide).

   Select or create a coordinate system. The SELECT AXIS menu appears with the options **X axis**, **Y axis** and **Z axis**. Choose one of them.

5. Select **Surface** or **Area** from the AREA menu.

6. Select a part face or a quilt, as appropriate, for projection. The system highlights the selected entity in red, and displays the projected area in the message window.

7. To determine the area projected by other faces or quilts in the same direction, repeat Step 6. To change the surface type, return to Step 5.

8. To re-define the direction of projection, return to Step 1.

   Note:

   If a surface intersects itself when it is projected, the system does not calculate its projected area and displays an appropriate error message.

Diameter

The Measure command Diameter measures the diameter of any revolved surface of a part, that is, a surface

created by revolving any type of sketched entity, or by extruding an arc/circle.

How to measure the diameter of a revolved feature

1. Choose **Diameter** from the MEASURE menu.

2. Select a surface to measure diameter. The surface selected highlights in red, and the diameter *at the pick point* is displays in the message window.

Draft

The command Draft is used to evaluate 2-dimensional draft geometry using Sec Info menu options. The command is only available in Drawing mode. Refer to 2-D Drafting in the *Drawing User's Guide* for more information.

Transform

The Measure command Transform generates a transformation file containing the transformation matrix values between two coordinate systems. This information is in the proper format to create other coordinate systems using the From File Datum coordinate system option (refer to Creating an Offset Coordinate System of the *Part Modeling User's Guide*).

How to measure the transform between two coordinate systems

1. Choose **Transform** from the MEASURE menu.

2. Select or create a coordinate system as the basis for the measurement.

3. In Assembly mode, the system prompts you to select a component.

4. Select or create a second coordinate system.

5. Enter the file name, "*filename*", in which to store the transform data. The system will automatically append the extension ".trf."

Thickness

The Measure command Thickness measures the thickness of a part to determine if a specified region has a thickness that is greater than or less than a user-specified maximum or minimum value. The region is displayed as a cross-section.

- If a region's thickness is *over* the maximum allowable, it displays with a *red* border.

- If a region's thickness is *below* the minimum allowable, it displays with a *blue* border.

Choosing **Thickness** causes the SETUP THK CK menu to appear with the options grouped as shown below.

Group 1:

- **Sel Plane**-Select or create one or more datum planes in which to check thickness.

- **Make Slices**-Define slices in which to check thickness. Using slices enables you to perform a thickness check for a large region of a model by dividing it into a series of parallel planes (slices) separated by a specified value (offset).

Group 2 (accessible only if you selected **Make Slices** above):

- **Start Point**-Pick the startpoint for defining slices.

- **End Point**-Pick the endpoint for defining slices.

- **Slice Dir**-Indicate the direction in which to create slices.

- **Slice Offset**-Specify the interval between slices.

Group 3:

- **MaxThickness**-Specify a value for the maximum allowable wall thickness.

- **MinThickness**-Specify a value for the minimum allowable wall thickness.

Group 4:

- **Done**-Execute the thickness check.

- **Quit**-Abort the thickness check.

When the thickness check is complete, the cross section highlights as follows:

- *Yellow*-section's thickness is between the specified maximum and minimum values.

- *Red border*-section's thickness exceeds the specified maximum value.

- *Blue border*-section's thickness is below the specified minimum value.

In addition, the THICK DISP menu appears with the following options:

- **Setup Thk Ck**-Return to the SETUP THK CK menu. (Choosing **Done** from the SETUP THK CHK menu then returns you to the THICK DISP menu)

- **Info**-Display Information Window with thickness check information as follows:

  first,

    ○ Current plane number on the part.

    ○ Specified maximum and minimum values.

then, information on the cross section(s) as follows:

  o Is the thickness of any part of the cross section over the maximum value (if specified)?

  o Is the thickness of any part of the cross section under the minimum value (if specified)?

  o The area of the cross section.

- **Next**-Display the next cross-section.

- **Previous**-Display the previous cross-section.

- **Go To**-Display a specified slice number.

- **All**-Display all generated cross-sections.

- **Clear**-Remove all cross-sections from the display.

- **Done/Return**-Continue the thickness check.

Using "Sel Plane"

Thickness Check Performed Using Sel Plane

Regions within the cross-section which violate the maximum thickness are highlighted in red.

The cross-section of the model is cross-hatched in yellow.
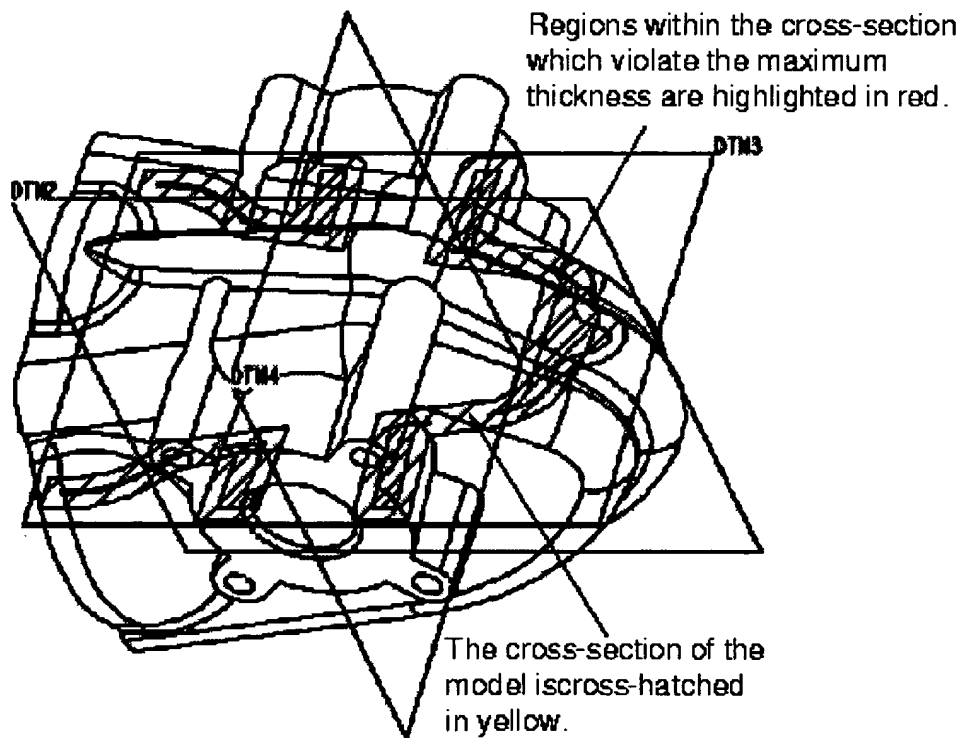
How to measure the thickness of a model using Sel Plane

1. Choose **Info, Measure, Thickness**. The SETUP THK CK menu appears.

2. Choose **Sel Plane** to select one or more planes for performing the thickness check, as well as

MaxThickness and/or **MinThickness** in order to provide values upon which to base the check.

3. Choose **Done** to initiate the thickness check.
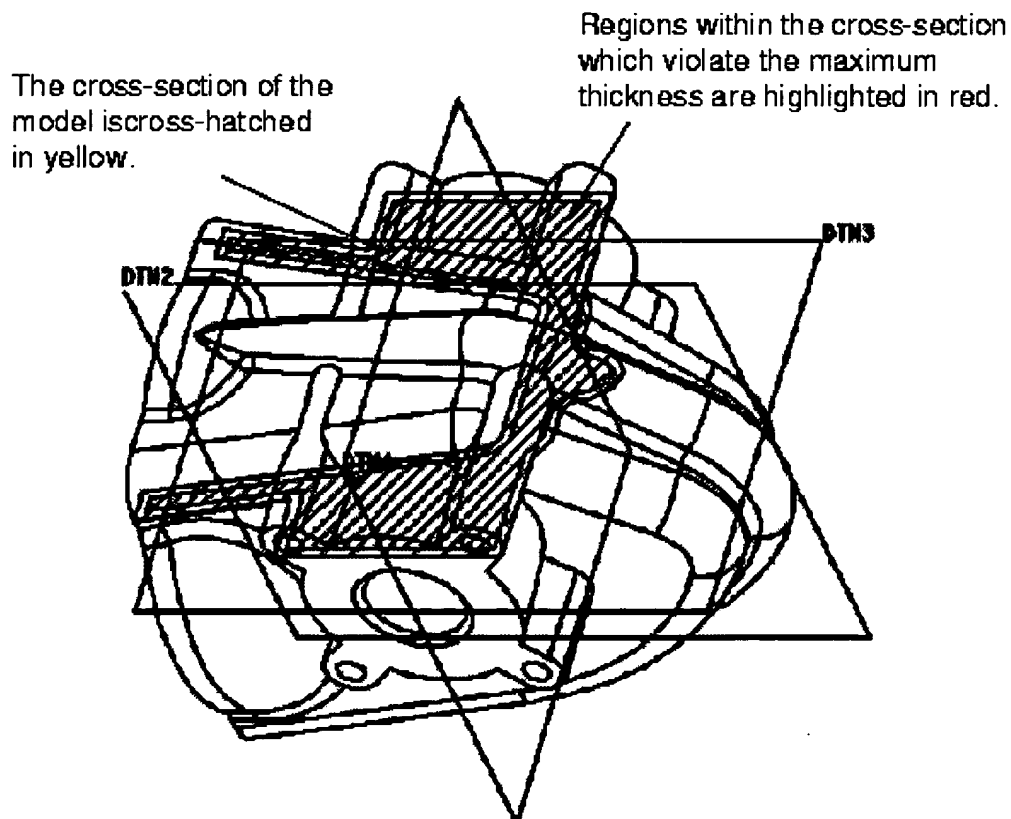
4. Pick the part on which to perform the check; the part highlights.

5. Pick or create one or more datum planes to be used in the thickness check, and choose **Done Plane** when you are finished.

6. If you chose **MaxThickness**, enter a value for the *maximum* allowable wall thickness.

7. If you chose **MinThickness**, enter a value for the *minimum* allowable wall thickness.

8. The cross-section of the part along the first selected plane is displayed (*see Thickness Check Performed Using Sel Plane for an example*), and the THICK DISP menu appears. Regions of the model that are *thicker* than the maximum allowed highlight in *red* and regions that are *thinner* than the minimum allowed highlight in *blue*.

9. If you selected more than one plane, you can use the **Next** and **Previous** commands or the **Go To** command (in the THICK DISP menu) to toggle from one cross-section to another. You can choose **All** to display all of the cross-sections at a single time, or choose **Clear** to remove all of the cross-sections (except the current one) from the display. Choosing the **Info** command brings up the Information Window, displaying summary information on the slices.

Using "Make Slices"

A Single Slice of a Thickness Check

The cross-section of the model iscross-hatched in yellow.

Regions within the cross-section which violate the maximum thickness are highlighted in red.

How to measure the thickness using Make Slices

1. Choose **Info, Measure, Thickness**. The SETUP THK CK menu appears.

2. Choose **Make Slices**; **Start Point, End Point, Slice Dir**, and **Slice Offset** become accessible and are chosen (checked off) automatically.

3. Choose **Done** to initiate the thickness check. The system prompts you to define the slices as follows:

4. Pick the part on which to perform the check; the part highlights.

5. On the part, pick a startpoint for defining the first slice, then pick the endpoint for defining the last slice. The line defined by these two points is extruded into a plane that is used as the first slice in the thickness check.

6. The GEN SEL DIR menu appears, with the commands **Plane, Crv/Edge/Axis, Csys**, and **Quit**. Pick an entity to which the direction of slice creation is normal.

7. An arrow appears, originating at the selected entity. Use **Flip** or **Okay** to indicate the direction of the thickness check, which is the direction in which to create the slices.

8. At the prompt, enter a value for the offset (interval) between slices.

9. At the prompt, enter a value for the *maximum* allowable wall thickness.

10. At the prompt, enter a value for the *minimum* allowable wall thickness.

11. A series of cross-sections are created, each separated from the other by the slice offset value (*see A Single Slice of a Thickness Check for an example of a slice*). The THICK DISP menu also appears.

    Notes:

    ■ Regions of the model that are *thicker* than the allowable maximum highlight in *red*.

    ■ Regions that are *thinner* than the minimum allowable thickness highlight in *blue*.

12. You can use the **Next** and **Previous** commands (in the THICK DISP menu) to toggle from one cross-section to another. You can choose **All** to display all of the cross-sections at a single time, or choose **Clear** to remove all of the cross-sections from the display. Choosing the **Info** command brings up the Information Window, displaying summary information on the slices.

Retrieving and Displaying Cross-Sections

You can select the edges of a cross-section for use in many MEASURE menu commands.

How to select the edges of a cross-section

1. Choose **Show Xsec** from the MEASURE menu to retrieve and display the cross-section.The

name-list menu XSEC NAMES appears.

2. Select one or more names from the namelist. Each retrieved cross-section displays in yellow. If any changes have been made to the model since the cross-section was created, the cross-section automatically updates according to those changes.

Each selected cross-section remains visible and selectable until you exit from the MEASURE menu.

Model Information

Feature Information

You can find information about features by choosing Feat Info from the Info menu. Examples are shown the following figures.

Typical Feature Information File (Protrusion)

```
PART NAME = PRT0001

FEATURE NUMBER       1
INTERNAL FEATURE ID  1
CHILDREN - 20(+2)

FIRST FEATURE: Extrude

NO. ELEMENT NAME    INFO                        STATUS
--- ------------- -----------                   ------
  1 Section                                     Defined
  2 Depth         Blind, depth - 5              Defined

SECTION NAME - S2D0026
FEATURE'S DIMENSIONS:
d0   10.00
d1 = 8.00
d2   5.00
```

Typical Feature Information File (Hole)

```
PART NAME - PRT0001

FEATURE NUMBER       2
INTERNAL FEATURE ID  20
PARENTS    1(#1)
TYPE = HOLE
FORM - EXTRUDED
DEPTH = FROM SKETCH TO ALL
PLACEMENT - LINEAR
FEATURE'S DIMENSIONS:
d3 = 2.00 Dia
d4 - 3.00
d5 = 2.00
```

After choosing Feat Info, you must specify a feature either by picking it with the mouse or using the Sel By Menu option. It is helpful to use the Feat Num option when the feature was unattached. Feat Info may

be chosen in both Part and Assembly modes. Feature information is shown in the INFO window and written to a file "feature.inf" in your current working directory. Outputting information about another feature overwrites the previous "feature.inf" file.

Feature List

When you choose Feature List, the Information Window appears with a table listing the features in order and giving information, such as Number, ID, Name, Type, Suppression Order and Regeneration Status (Regenerated, Unregenerated, Failed etc.). An example is shown below.

Typical Feature List

```
MODEL NAME        : BLOCK
FEATURE LINK LIST:
* * * * * * * * * * * * * * * * *


Num  ID      Name      Type           Sup Order   Regen Status

0001 000001 DATUM_1   DATUM PLANE                 Regenerated
0002 000004 DATUM_2   DATUM PLANE                 Regenerated
0003 000006 DATUM_3   DATUM PLANE                 Regenerated
0004 000008 BRICK     PROTRUSION                  Regenerated
0005 000029 VERT_HOLE HOLE                        Regenerated
0006 000081           ROUND                       Regenerated
0007 000111           CUT                         Regenerated
0008 000135           SURFACE                     Regenerated
     000180           COSMETIC                    Regenerated
```

Part Information

You can access information about every feature on a part by choosing Model Info.

- In Part mode, the Info window appears immediately.

- In Assembly mode, choose Part from the Model Info menu and specify a part, either by picking a part with the mouse (Pick) or entering the name of the part (Name).

Both regular and suppressed features are listed. In addition, all coordinate systems, cross sections, and reference dimensions are listed. Parts are labeled if they are a generic or an instance.

Note:

> In order to use the Name option on a part, the part must have been created or retrieved during the working session.

Assembly Information

You can access assembly information by choosing Model Info, then Assembly from the Info menu and

then either picking on an assembly or specifying the name of the assembly, or choosing Top Level to get information about the current assembly. An information window appears displaying the assembly information. The names of the components of the assembly display in a hierarchical structure to show how it was assembled. An example file appears below.

Sample Assembly Information File

```
ASSEMBLY NAME = PLATE

    LEVEL 1 PLATE ASSEM
      LEVEL 2 BOX PART
      LEVEL 2 CYL PART
      LEVEL 2 RING PART
      LEVEL 2 SUB_PLATE ASSEM
        LEVEL 3 DM PART
        LEVEL 3 BOLT PART
        LEVEL 3 BOLT PART
        LEVEL 3 BOLT PART
        LEVEL 3 BOLT PART

COMPONENTS:

COMPONENT NUMBER 1
INTERNAL COMPONENT ID  33
. . . . . . . . . . . . . . . . . . . . . . . . . .
```

Assembly information is discussed in greater detail in <u>Other Assembly Functions</u> in the *Assembly Modeling User's Guide*.

Full Path Name of Object

By default, the system lists only the *names* of the objects in the Model Info window. See the following example.

Model Info Config File Option "display_full_object_path" Disabled.

```
ASSEMBLY NAME = TEST_ASSY

    LEVEL 1 TEST_ASSY ASSEM
      LEVEL 2 BASE_ASSY ASSEM
        LEVEL 3 BASEPLATE PART
        LEVEL 3 BUSHING PART
        LEVEL 3 SNAPRING PART
      LEVEL 2 BASEPLATE PART
      LEVEL 2 BASEPLATE_1 PART
      LEVEL 2 BASEPLATE_1 PART
```

However, if you set the configuration-file option"display_full_object_path" to "yes," the system displays the *full pathname* of the objects, along with their *object-type* and *version-number* suffices. See the example below.

Model Info Config File Option "display_full_object_path" Enabled.

```
ASSEMBLY NAME = TEST_ASSY/dublin/Daedalus/testpro/test_assy.asm.15

LEVEL 1 TEST_ASSY ASSEN/dublin/Daedalus/testpro/test_assy.asm.15
LEVEL 2 BASE_ASSY ASSEN/dublin/Daedalus/testpro/base_assy.asm.75
LEVEL 3 BASEPLATE PART/dublin/Daedalus/testpro/baseplate.prt.40
LEVEL 3 BUSHING PART/dublin/Daedalus/testpro/bushing.prt
LEVEL 3 SNAPRING PART/dublin/Daedalus/testpro/snapring.prt.56
LEVEL 2 BASEPLATE PART/dublin/Daedalus/testpro/baseplate.prt.40
LEVEL 2 BASEPLATE_1 PART/dublin/Daedalus/testpro/baseplate.prt.40
LEVEL 2 BASEPLATE_1 PART/dublin/Daedalus/testpro/baseplate.prt.40
```

Parent/Child Information

The ParentChild option in the Info menu is used to highlight the relationships between features. If you select either option Parents or option Children, you can then choose either to create an information file or to highlight the appropriate geometry on the screen. If you choose the file option, the information is written to a file *and* displays in the Information Window.

The Parent/Child menu options are:

- **Parents**-Shows all the parents for the selected feature. The parent features highlight in the Reference Color (typically light blue).

- **Children**-Shows all the children for the selected feature. The child features highlight in the Reference Color.

- **References**-Shows, one at a time, all the references used to construct a feature. These can be axes, datums, surfaces, edges or other features. They are shown in the Internal Reference color. Pro/ENGINEER also displays information about the references in the message window in a form similar to "Reference No. 2 (id = 24) of feature No. 3 (id = 31)"

  Note:

  If a reference feature is suppressed, Pro/ENGINEER displays an asterisk in place of its feature number.

- **Child Ref**-Shows the references for each and highlights in the Surface Mesh color (typically dark blue) and the surfaces, edges or points which they reference highlight in the Internal Quilt Edge color (typically magenta).

How to show Parent/Child relationships of a particular feature

1. Choose **ParentChild** from the INFO menu.

2. Choose one of the options from the PARENT/CHLD menu.

3. If you chose **Parents** or **Children**, choose one of the options from the FILE/HILITE menu:

   o **File**-A text file with the corresponding feature IDs listed is written to disk and displays in the

Information Window. The file name displays at the bottom of the Information Window.

- ○ **Highlight**-The appropriate geometry highlights with the color codes given above.

  Select the feature.

4. If you chose **References**, select the feature. The SHOW REF menu appears. Choose one of the following.

   - ○ **Next** or **Previous**-Select the next or previous reference.

   - ○ **Info**-Displays information about the highlighted reference in the Information Window.

5. If you chose **Child Ref**, the CHILD REFS menu appears. Choose one of the following.

   - ○ **Next** or **Previous**-Select the next or previous reference.

   - ○ **Ref Info**-Displays information about the highlighted reference in the Information Window.

   - ○ **Child Info**-Displays information about the child feature(s), using the highlighted reference in the Information Window.

Bill of Materials

A bill of materials (BOM) is a listing of all parts and part parameters in the current assembly. You can customize the output format to produce a particular form of presentation and content. BOMs can be created for assemblies in Assembly mode or from assembly drawings in Drawing mode.

The information in this section explains how to create and format simple text BOMs, which are stored as text files. The optional module Pro/REPORT provides functionality for creating BOM reports: graphical BOMs with complex formatting and indexing. Refer to <u>Reports</u> of the Drawing User's Guide for information on creating and using BOM reports.

The source of the bill of materials (BOM) output format can be configured by the configuration file. An example of the configuration file option for a user-defined format is:

bom_format *mybomstyle*.fmt

The default output format for the BOM is divided into two sections:

- *breakdown*-lists the name, type, and number of instances of each member and sub-member.

- *summary*-lists the total quantity of each part included in the assembly. It amounts to a "shopping list" of all the parts needed to build the assembly from the part level.

A user-defined BOM output format specifies the format of the breakdown section and the summary section separately. You can include one or both sections, but you must specify the column titles, row content, and display format for each included section.

The user-defined format file contains a combination of text and formatting commands. The overall layout

of the BOM is specified with the following section commands which appear at the beginning of a line:

- **.breakdown**-This command appears at the beginning of the breakdown section of the BOM. Lines of text following the ".breakdown" command appear as a heading for each assembly breakdown.

- **.summary**-This command appears at the beginning of the summary section of the BOM. Lines of text following the ".summary" command appear as a heading for the summary section.

- **.titles**, **.row**-These commands may be used within each section of the BOM to specify the column titles and the information to be included on the repeating rows of the BOM.

At any point in the format file, you can specify a system-supplied or a user-defined attribute:

**%$**-A percent sign followed by a dollar sign indicates that the next word is one of the three system-supplied attributes: name, type, or quantity. Example: %$type.

%-Any word preceded by a percent sign only is a user-defined parameter.

Text which is not a part of a parameter name is printed as it appears. Example: to print a dollar sign before the user-defined attribute *price*, type $%price.

System-supplied and user-defined parameters may be followed by a
C language-style format specifier, enclosed in square brackets ([ ]), that indicates the data type and printout width of the attribute's value. These format specifiers are optional, but may be needed for correct column alignment. Examples of format specifiers are as follows:

[4s]-The preceding parameter is a string variable (text), and the field width is 4.

[3d]-The preceding parameter is an integer, and the field width is 3.

[6.2f]-The preceding parameter is a floating point variable (real number). The total field width is 6 (including a decimal point), and there are 2 digits after the decimal point. Example: 400.25.

Field widths may be overridden by strings which are longer than the specified field. To ensure that the field remains the proper size, use the following format:

[8.8s]-The preceding parameter is a string variable. The field width is 8, and longer strings is truncated to 8 characters.

To left-justify the text for a field in each row, use a minus sign before the number:

[-4s]-The text in this field is left-justified.

In each of the above format specifiers, the letter is optional; Pro/ENGINEER determines the type of the user-defined parameter (string, integer, or floating point) if you omit it.

Entering the field [%[$total(item)]] in the format file (where "item" may be a user-defined parameter name) calculates the total for "item." When entered within a ".row" command, the total is for that row only. Anywhere else, the "$total()" command must also be preceded by the ".row" command and the total is for all "item" within the sub-assembly or assembly (*see the example in the section Bill of Materials*). You can also use [%[$total($quantity)]], where "quantity" is a system-supplied attribute.

Note:

In the "$total(item)" command, don't precede "item" or "$quantity" by `%'.

The system checks the format file for errors. If an error is found, the BOM format file appears in an Information Window with the error highlighted.

Examples

The model helicopter engine shown below is the assembly which has been used for the BOM examples which follow.

Model Helicopter Engine Assembly
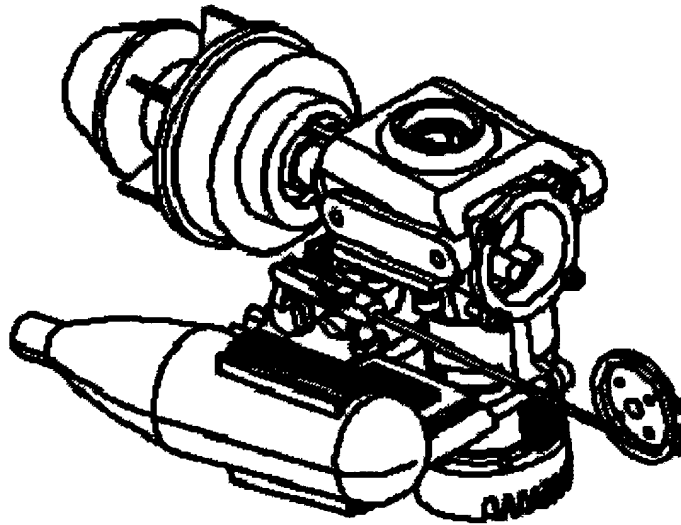
The default BOM format file is shown below:

```
.breakdown

%$type %$name contains:

.row %$quantity[6d]; %$type[12s]; %$name

.summary

Summary of parts for assembly %$name

.row %$quantity[6d]; %$type[12s]; %$name
```

The BOM that this default format generates follows:

```
Assembly ENGINE_MAIN_ENGINE contains:

1 Part ENG_LAYOUT

1 Part ENG_MOUNT
```

1 Sub-Assembly ENG_ENGINE

1 Sub-Assembly ENG_STARTER

1 Sub-Assembly THROTTLE

1 Sub-Assembly THRTL_PUSHROD

1 Part SERVO_DISC

Sub-Assembly ENG_ENGINE contains:

1 Part ENG_CRANK_LAYOUT

1 Part ENG_BLOCK

1 Part ENG_SHAFT

1 Part ENG_ROD

1 Part ENG_PISTON

1 Part ENG_CAP

1 Part ENG_SLEEVE

1 Part ENG_HEAD

1 Part ENG_MUFFLER

1 Part ENG_PLUG

1 Part GAS_NIPPLE

10 Part BOLT_2X8

2 Part BOLT_3X38

6 Part BOLT_3X8

Sub-Assembly ENG_STARTER contains:

1 Part ENG_CLUTCH

1 Part ENG_START

1 Sub-Assembly BRNG_18X4

1 Part ENG_GEAR

Sub-Assembly BRNG_18X4 contains:

1 Part BEARING_OR_18X4

Sub-Assembly THROTTLE contains:

1 Part THRTL_BLOCK

1 Part THRTL_SCREW

1 Part THRTL_ACTUATOR

1 Part THRT_SPRING

1 Part THRTL_KNOB

1 Part THRTL_NEEDLE

1 Part NUT_5

1 Part GAS_NIPPLE

1 Part NUT_3

1 Part BOLT_3X8

Sub-Assembly THRTL_PUSHROD contains:

1 Part THRTL_PUSHROD

1 Part THRTL_ROD_END

Summary of parts for assembly ENGINE_MAIN_ENGINE:

1 Part ENG_LAYOUT

1 Part ENG_MOUNT

1 Part ENG_CRANK_LAYOUT

1 Part ENG_BLOCK

1 Part ENG_SHAFT

1 Part ENG_ROD

1 Part ENG_PISTON

1 Part ENG_CAP

1 Part ENG_SLEEVE

1 Part ENG_HEAD

1 Part ENG_MUFFLER

1 Part ENG_PLUG

2 Part GAS_NIPPLE

10 Part BOLT_2X8

2 Part BOLT_3X38

```
7 Part BOLT_3X8

1 Part ENG_CLUTCH

1 Part ENG_START

1 Part BEARING_OR_18X4

1 Part ENG_GEAR

1 Part THRTL_BLOCK

1 Part THRTL_SCREW

1 Part THRTL_ACTUATOR

1 Part THRT_SPRING

1 Part THRTL_KNOB

1 Part THRTL_NEEDLE

1 Part NUT_5

1 Part NUT_3

1 Part THRTL_PUSHROD

1 Part THRTL_ROD_END

1 Part SERVO_DISC
```

How to use the default BOM output format

1. Choose **Info** from the MAIN menu.

2. Choose **BOM** from the INFO menu. The BOM displays in an Information Window and written to file.

An example of a customized BOM format file appears below. Note the use of spaces and format specifiers to produce the columns in the resulting BOM file. Lines cannot be continued on a following line.

Note:

In order to fit the following example into the page format of this document, there are instances below in which a single line of text in the BOM format file appears in a smaller font.

```
.breakdown

%$type %$name contains:

.titles Qty ;   Name ;   Type ; Manufacturer ;   Cost ; Availability;

---------------------------------------------------------------------
```

```
.row %$quantity[-5d]; %$name[18.18s]; %$type[-9.9s];>
   %manufacturer[-12.12s];%cost[-4.2f];%availability[10.10s]
===================================================================
```

.summary

Summary of parts for assembly %$name:

.titles Quantity;   Name;

```
-----------------------------------------------------------------
```

.row %$quantity[-5d] ; %$name[-20.20s]

TOTAL COST: $%[$total(cost)][4.2f]

The BOM which is generated from this format is shown below:

Assembly ENGINE_MAIN_ENGINE contains:

Qty	Name	Type	Manufacturer	Cost	Availability
1	ENG_LAYOUT	Part	N/A: datum		
1	ENG_MOUNT	Part	Northeast	0.85	Aug. 92
1	ENG_ENGINE	Sub-Assem			
1	ENG_STARTER	Sub-Assem			
1	THROTTLE	Sub-Assem			
1	THRTL_PUSHROD	Sub-Assem			
1	SERVO_DISC	Part	Precision	0.10	Aug. 92

```
===================================================================
```

Sub-Assembly ENG_ENGINE contains:

Qty	Name	Type	Manufacturer	Cost	Availability
1	ENG_CRANK_LAY	Part	N/A: datum		
1	ENG_BLOCK	Part	Northeast	0.97	Aug. 92
1	ENG_SHAFT	Part	Acme	0.70	Sept. 92
1	ENG_ROD	Part	Local Steel	0.17	June 92
1	ENG_PISTON	Part	In-house	0.00	July 92
1	ENG_CAP	Part	Local Steel	0.21	Aug. 92
1	ENG_SLEEVE	Part	In-house	0.00	June 92
1	ENG_HEAD	Part	Northeast	0.32	Oct. 92
1	ENG_MUFFLER	Part	In-house	0.00	May 92
1	ENG_PLUG	Part	Wilder	0.08	April 92
1	GAS_NIPPLE	Part	Wilder	0.05	April 92

Qty	Name	Type	Manufacturer	Cost	Availability
10	BOLT_2X8	Part	Standard	0.06	April 92
2	BOLT_3X38	Part	Standard	0.10	April 92
6	BOLT_3X8	Part	Standard	0.07	April 92

==

Sub-Assembly ENG_STARTER contains:

Qty	Name	Type	Manufacturer	Cost	Availability
1	ENG_CLUTCH	Part	Acme	0.60	July 92
1	ENG_START	Part	Precision	1.10	May 92
1	BRNG_18X4	Sub-Assem			
1	ENG_GEAR	Part	Precision	0.25	June 92

==

Sub-Assembly BRNG_18X4 contains:

Qty	Name	Type	Manufacturer	Cost	Availability
1	BEARING_OR_18X4	Part	Standard	0.09	April 92

==

Sub-Assembly THROTTLE contains:

Qty	Name	Type	Manufacturer	Cost	Availability
1	THRTL_BLOCK	Part	Northeast	2.10	Nov. 92
1	THRTL_SCREW	Part	Acme	0.33	May 92
1	THRTL_ACTUATOR	Part	Acme	0.22	June 92
1	THRT_SPRING	Part	In-house	0.00	April 92
1	THRTL_KNOB	Part	Precision	0.23	Aug. 92
1	THRTL_NEEDLE	Part	Wilder	0.35	April 92
1	NUT_5	Part	Standard	0.07	April 92
1	GAS_NIPPLE	Part	Wilder	0.05	April 92
1	NUT_3	Part	Standard	0.05	April 92
1	BOLT_3X8	Part	Standard	0.07	April 92

==

Sub-Assembly THRTL_PUSHROD contains:

Qty	Name	Type	Manufacturer	Cost	Availability
1	THRTL_PUSHROD	Part	In-house	0.00	May 92

```
      1       THRTL_ROD_END    Part     In-house          0.00      June 92

      ==========================================================

      Summary of parts for assembly ENGINE_MAIN_ENGINE:

      Qty     Name

      -----------------------------------------------------------------

      1       ENG_LAYOUT

      1       ENG_MOUNT

         1    ENG_CRANK_LAYOUT

         1    ENG_BLOCK

         1    ENG_SHAFT

         1    ENG_ROD

         1    ENG_PISTON

         1    ENG_CAP

         1    ENG_SLEEVE

         1    ENG_HEAD

         1    ENG_MUFFLER

         1    ENG_PLUG

         2    GAS_NIPPLE

        10    BOLT_2X8

         2    BOLT_3X38

         7    BOLT_3X8

         1    ENG_CLUTCH

         1    ENG_START

         1    BEARING_OR_18X4

         1    ENG_GEAR

         1    THRTL_BLOCK

         1    THRTL_SCREW

         1    THRTL_ACTUATOR

         1    THRT_SPRING

         1    THRTL_KNOB

         1    THRTL_NEEDLE

         1    NUT_5

         1    NUT_3

         1    THRTL_PUSHROD
```

```
1    THRTL_ROD_END

1    SERVO_DISC
```
TOTAL COST: $10.18

How to use a user-defined BOM output format file

1. Using the system editor, create the BOM output format file.

2. Add the following option to the configuration file:

    bom_format *formatname*.fmt

3. Within Pro/ENGINEER, add user-defined parameters to parts and assemblies using the RELATIONS menu option **Add Param**.

4. In Assembly mode, choose **Info** from the MAIN menu; in Drawing mode, choose **Info** from the DRAWING menu.

5. If you are creating a BOM for an assembly that is a member of a family of assemblies, the MODEL TYPE menu appear, with the options **Generic** and **Instance**. Choose **Generic** to create a BOM for the generic assembly, and choose **Instance** to create a BOM for the instance of the assembly.

6. Select **BOM** from the INFO menu. The BOM displays in an Information Window and written to file.

   Note:

   Assembly members that are blanked on a layer or suppressed through assembly representation are listed in the BOM for the assembly as if they were displayed in the normal way.

Audit Trails

Choosing Audit Trail from the Info menu displays revision information for the current object if it has been fetched from a product database through Pro/PDM.

The following figure shows the format of an audit trail
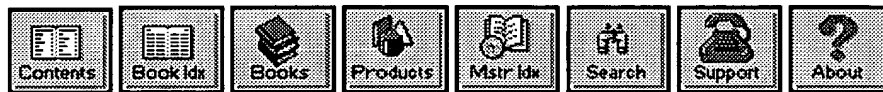
Format of Audit Trail

Mod Time	Rev	Rel Level	User	Action	Comment

Each audit trail displays the following information:

* Mod Time-The date and time of the most recent modification to the object, in the format mm/dd/yy hh:mm:ss.

- Rev-The object's current revision in its product database of origin (for example: 1.1).

- Rel Level-The object's current release level in its product database of origin.

- User-The owner of the object in its product database.

- Action-A list of actions that have been taken on the object, including fetching, storing, renaming, and copying.

- Comment-Comments associated with the object during submission, approval, rejection, etc.

Note that the configuration file option "info_output_mode" determines the means by which the system outputs the audit trails.